HEIDENHAIN

TNC 320

The Compact Contouring Control for Milling, Drilling and Boring Machines

September 2014

This brochure describes the functions and specifications of the TNC 320 with NC software 771851-01.

Contents

The TNC 320...

Where can it be used?	Compact and versatile – The right control for milling, drilling and boring machines	4
What does it look like?	Well designed and user friendly – The TNC 320 in dialog with the user – The functional user interface	6
What can it do?	Minimize setup and nonmachining time – The TNC 320 makes setup easy – The TNC 320 manages and measures	10
	Economical and productive – Swivel head or rotary table controlled by the TNC – Machining any contour slots with trochoidal milling	14
How is it programmed?	Programming, editing, testing – The TNC 320 opens endless possibilities – Graphic support in any situation	16
	Programming in the workshop – Straightforward function keys for complex contours – Programming contours unconventionally – Field-proven cycles for recurring operations – Reusing programmed contour elements – Fast availability of all information	18
	Open for communication – The TNC 320 understands DXF files – Fast data transfer with the TNC 320 – Display any data formats on the TNC screen – The TNC 320 programming station	24
Are there any accessories?	Workpiece measurement – Setup, presetting and measuring with touch trigger probes	29
	Tool measurement – Measuring length, radius and wear right on the machine	30
	Positioning with the electronic handwheel – Delicate axis traverse	31
... at a glance	Overview – User functions – Options – Accessories – Specifications	32

Compact and versatile
– The right control for milling, drilling and boring machines

For more than 30 years, TNC controls from HEIDENHAIN have been proving themselves in daily use on milling, drilling and boring machines, and machining centers. While the controls have undergone continuous development during this period, the basic operational technique has remained the same.

You will find these principles implemented in the TNC 320 as well: shop-oriented programmability with graphic support, many field-proven cycles and an operational design familiar from other HEIDENHAIN controls.

Shop-oriented programming
You program conventional milling and drilling operations yourself at the machine, in **plain language dialog**—the workshop-oriented programming language from HEIDENHAIN. The TNC 320 provides you with optimum support with practical prompts, questions and graphical aids.

Standard operations and even complex applications are on call as a large variety of real-world machining cycles or coordinate transformations.

Easy to operate
For simple work, such as face milling, you need not write a program on the TNC 320. It is just as easy to operate the machine manually by pressing the axis keys or—for maximum sensitivity—using an electronic handwheel.

Offline program creation
The TNC 320 can be programmed remotely just as well. Its Fast Ethernet interface guarantees very short transfer times, even of long programs.

The TNC 320 is compact and easy to read.
The TNC 320 is a compact but versatile contouring control for three servo axes and servo spindle. Two further servo axes are an option. Thanks to its flexible operation—workshop-oriented programmability with HEIDENHAIN conversational programming or offline programming—and its scope of features, it is especially suited for use on universal milling, drilling and boring machines for the following:

- Series and single-part production
- Tool making
- Machine building
- Research and development
- Prototypes and pilot plants
- Repair departments
- Training and education facilities

It also offers the applicable features both necessary and helpful for:

Universal milling machines

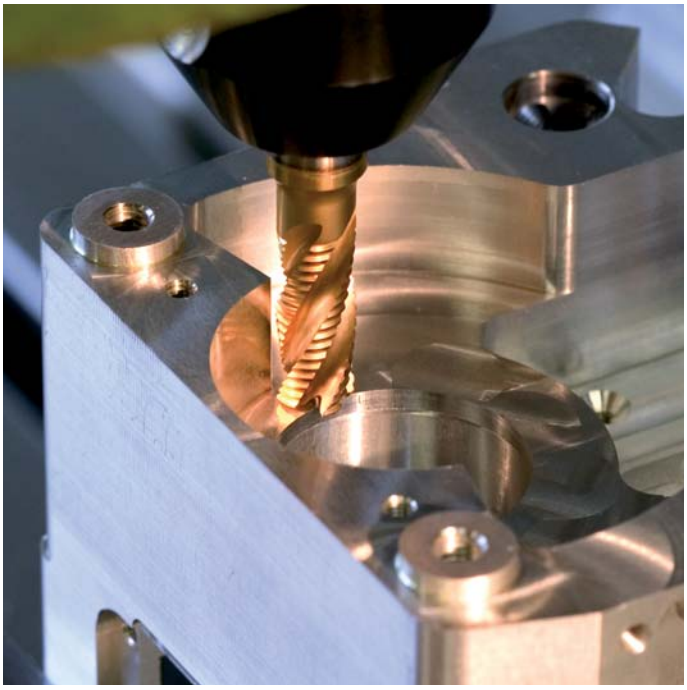
- Free contour programming
- Milling cycles for complex contours
- Fast presetting with HEIDENHAIN touch probes

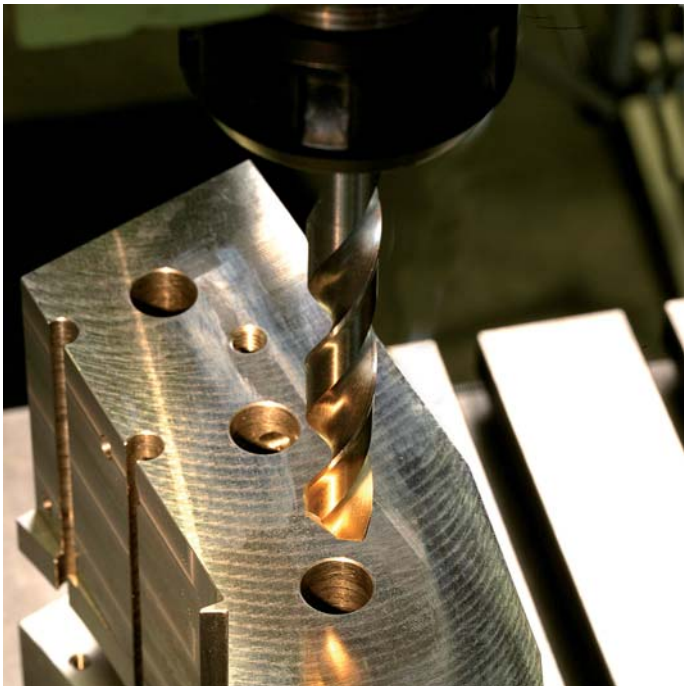
Drilling and boring machines

- Cycles for drilling, boring and spindle alignment
- Cycles for Cartesian and polar point patterns
- Drilling oblique holes

Machines with parallel secondary axes

- Compensating movement in the secondary axes U, V, W through the principal axes X, Y, Z
- Defining the principal and secondary axes in the NC program makes it possible to run programs on different machine configurations
- Including movements of the parallel axis in the position display of the associated principal axis (sum display)





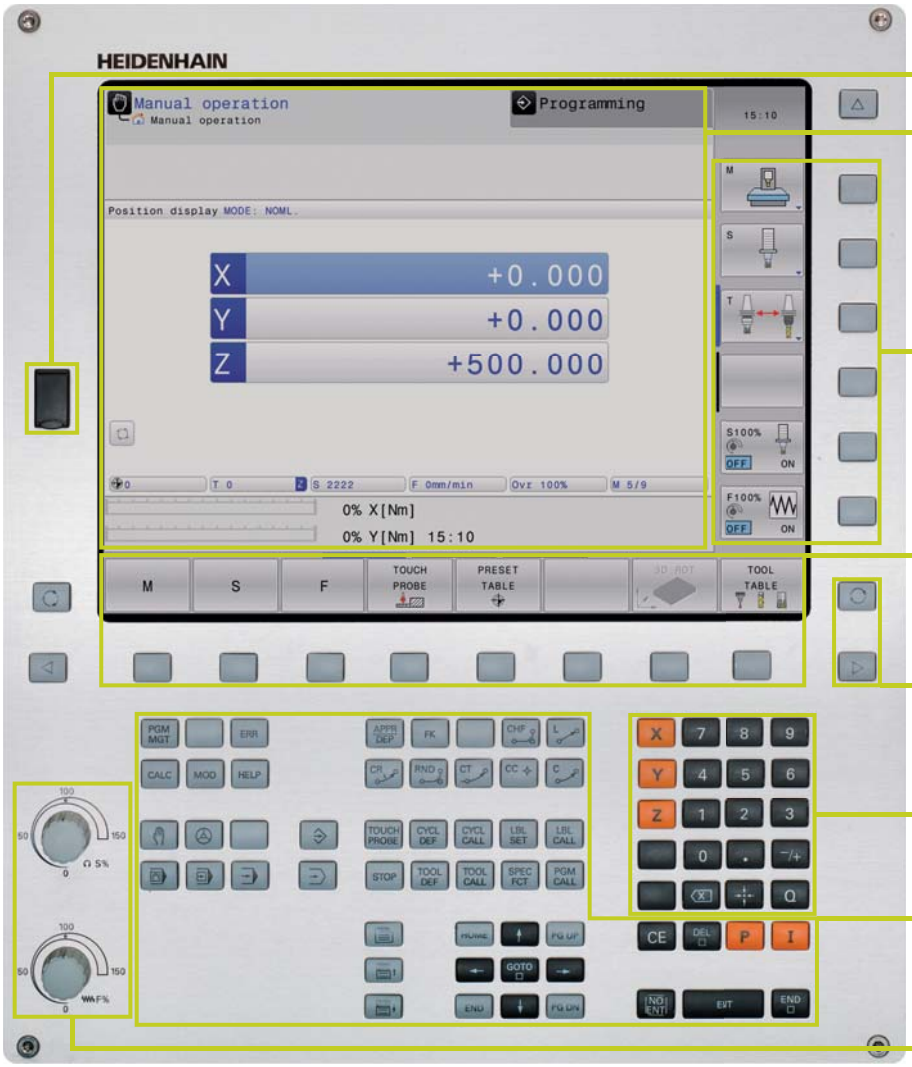
Well designed and user friendly
– The TNC 320 in dialog with the user

The screen
The large 15-inch TFT color flat-panel display shows a clear overview of all relevant information for programming, operating and inspecting the machine tool and control, such as program blocks, comments and error messages. More information is provided through graphic support during program entry, test run and actual machining.

The selectable split-screen display shows the part program blocks in one half of the screen and the graphics or the status display in the other half.

During the course of the program, status displays will always offer information to keep you up to date on tool position, the current program, active cycles and coordinate transformations, and other data. The TNC 320 even shows the current machining time.

The keyboard
As with all TNCs from HEIDENHAIN, the keyboard is tailored to the programming process. The well-thought-out arrangement of keys in a clear division into **function groups**, i.e. programming modes, machining modes, management/TNC functions and navigation, supports you during program input. Simple key assignment, easily understandable symbols or abbreviations clearly indicate each key's function. You use the **override potentiometers** to make delicate adjustments of feed-rate and spindle speed.



USB port for additional data storage or pointing devices

The screen content includes two operating modes, the program, graphics and the machine status

PLC function keys (soft keys) for machine functions

Self-explanatory **function keys** (soft keys) for NC programming

Keys for **screen management** (screen layout), mode of operation and for shifting between soft-key rows

Axis-selection keys and **numeric keypad**

Function keys for programming modes, machine modes, TNC functions, management and navigation

Override potentiometers for feed rate and spindle speed

Ergonomic and elegant, state-of-the-art and field-proven—HEIDENHAIN controls in a new design. Judge for yourself:

Durable
The high-quality stainless steel design of the TNC 320 features a special protective coating and is therefore highly resistant to soiling and wear.

Smooth
The rectangular, slightly rounded keys are pleasant to the touch and reliable in operation. Their inscriptions do not wear off, even under extreme workshop conditions.

Versatile
Soft keys both for the programming and the machine functions always show only the currently available selections.

Sensitive
With the handy control knobs you can individually adjust the feed-rate and spindle speed.

Communicative
The fast USB 2.0 interface lets you connect storage media or pointing devices to the keyboard simply and directly.

Flexible
The optional machine operating panel features easily exchangeable snap-on keys.

Reliable
The elevated key bed of the machine operating panel (with the version with a separate keyboard unit) prevents accidental actuation. LEDs serve for status display of each key by clearly indicating the active machine functions.

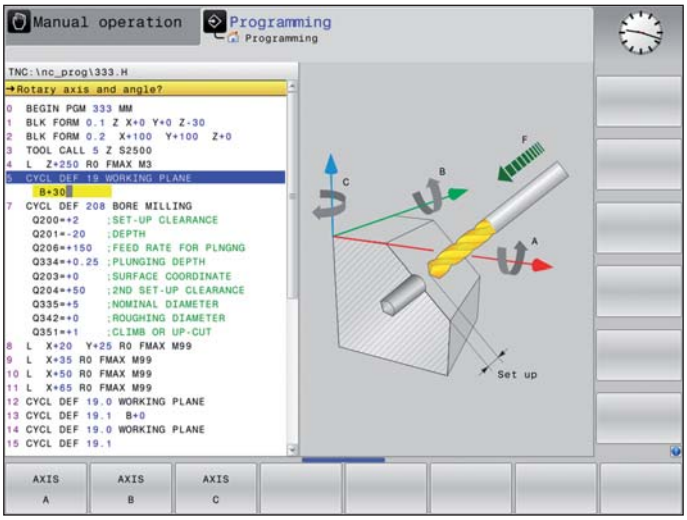
Well designed and user friendly
– The functional user interface

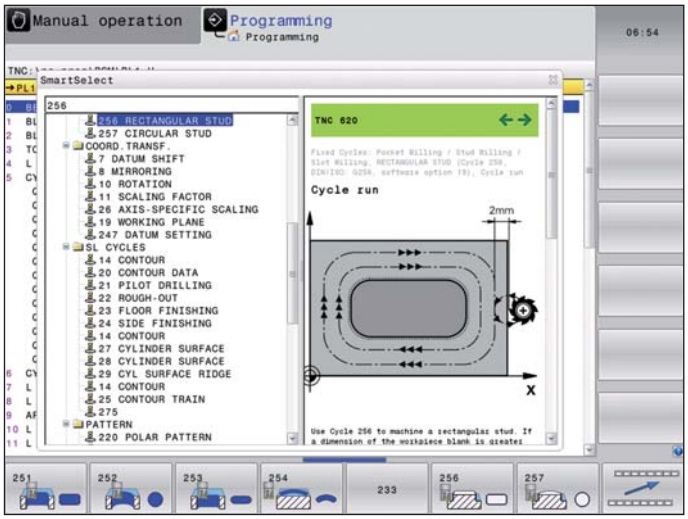
The combination of the straightforward and ergonomically designed keyboard and the well-designed screen layout are the essence of reliable and fatigue-free operation. These are principles that HEIDENHAIN controls have always embodied. However, the TNC 320 also offers a number of features that make working with the control even easier and user-friendlier than ever.

Attractive view
The user interface of the TNC 320 has a modern appearance, with lightly rounded forms, color gradients and a homogeneously designed font. The individual screen areas are clearly distinguished and the operating modes are also indicated by their respective symbols.

To better distinguish between the priority of error messages, the TNC 320 displays them in color-coded categories. A color-coded warning triangle is also displayed.

Fast function overview
With **smartSelect** you enjoy dialog guidance for selecting functions quickly and easily that up to now were accessible only through the soft-key structure. As soon as you open smartSelect, it displays a tree structure with all subordinate functions that can be defined in the control's current condition. Moreover, in the right part of the smartSelect window, the TNC displays the integrated help. With the cursor or a mouse click, you immediately access detailed information on the respective function. smartSelect enables you to define fixed cycles, touch probe cycles, special functions (SPEC FCT), and quickly access the parameter programming.



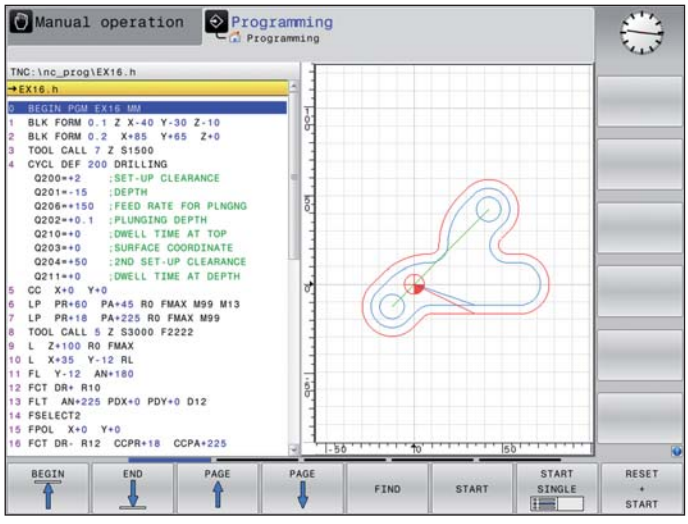


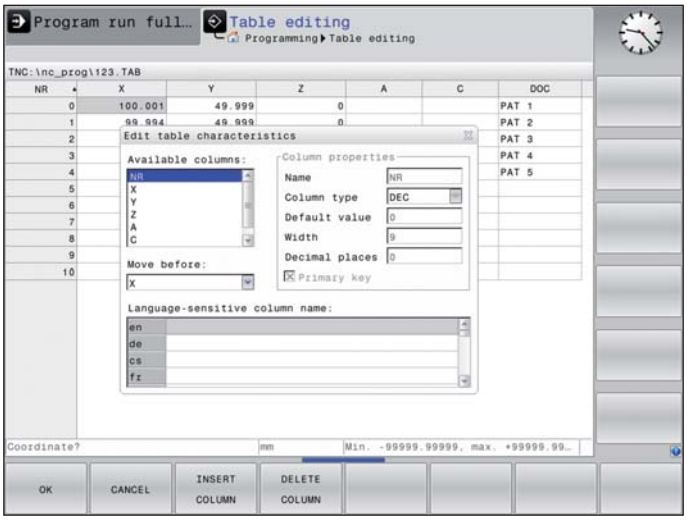
Color-structured programs
The content of a program line can be quite comprehensive: line number, program function, input values, comment. To help you always find your way even in complex programs, the individual program elements on the TNC 320 are shown in different colors. The color syntax highlighting improves your overview when editing NC programs. It enables you to see at a glance, for example, where the editable input values are.

Uniform table editor
Regardless of which table you are editing—whether the tool table, datum table or pallet table—the function and operation of the table editor are always the same.

Info line
In the info line, the TNC 320 shows the respective submode condition and helps you to orient yourself. The function is comparable with the history function in web browsers.

MOD function
The additional mode MOD offers a myriad of possible settings in a standardized layout regardless of the operating mode.





Minimize setup and nonmachining time
– The TNC 320 makes setup easy

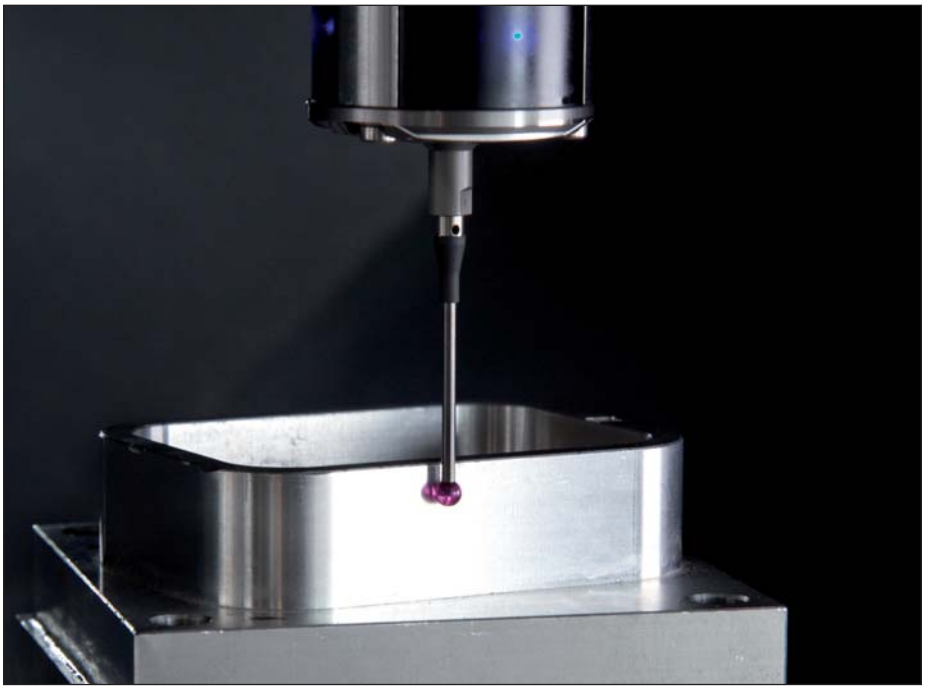
Before you can begin machining, you must first clamp the tool and set up the machine, find the position and orient the workpiece on the machine, and set the workpiece reference point. Without support from the control this is often a time-consuming procedure, but it is indispensable. After all, any error directly reduces the machining accuracy. Particularly in small and medium-sized production runs, as well as for very large workpieces, setup times become quite a significant factor.

Here the TNC 320 shows its strengths: With its practice-oriented setup features it supports the operator and helps to reduce nonmachining time. Together with the **touch probes,** the TNC 320 offers various probing features for aligning, presetting, and measuring the workpieces.

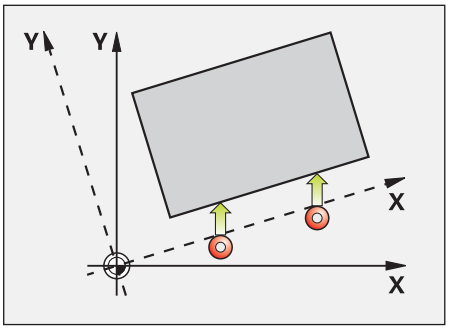
Delicate manual traverse
For setup, you can use the direction keys to move the machine axes manually or in incremental jog. A simpler and more reliable way, however, is to use the electronic handwheels from HEIDENHAIN (see page 31). Particularly with the portable handwheels you are always close to the action, enjoy a close-up view of the setup process, and can control the infeed responsively and precisely.

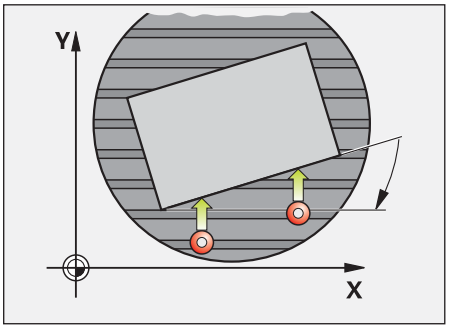
Workpiece alignment
With HEIDENHAIN touch probes (see page 29) and the probing functions of the TNC 320, you can forgo any tedious manual alignment of the workpiece:

- Clamp the workpiece in any position.
- The touch probe determines the actual workpiece position by probing a surface, two holes, or two studs.
- The TNC 320 compensates the misalignment with a "basic rotation," which means that in the NC program the part is rotated by the measured misalignment.



Compensating workpiece misalignment
Compensate misalignment by rotating the coordinate system or turning the table





Setting datums
You can use a reference point to assign a defined value in the TNC display to any workpiece position. Finding this point quickly and reliably reduces nonproductive time and increases machining accuracy.

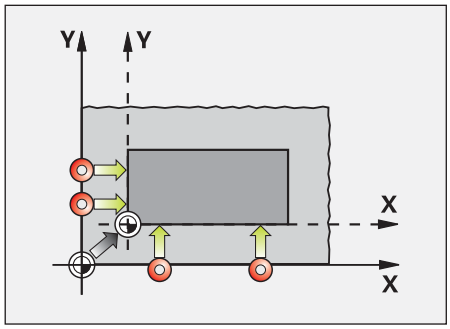
The TNC 320 features probing cycles for automatic presetting. Once found, you can save these datums
- in the datum management,
- in a datum table, or
- by directly setting the displayed value.

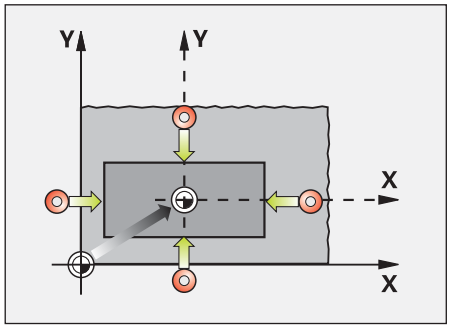
Datum management
The datum management makes flexible machining, shorter setup times and increased productivity possible. In other words, it makes it much easier to set up the machine.

In the datum management you can save **any number of datums** and assign an individual basic rotation to each one.

There are three ways to save datums rapidly in the datum management:
- In the Manual mode by soft key
- By using the probing functions
- With the automatic probing cycles

Set datum
At a corner, for example, or in the center of a circular stud



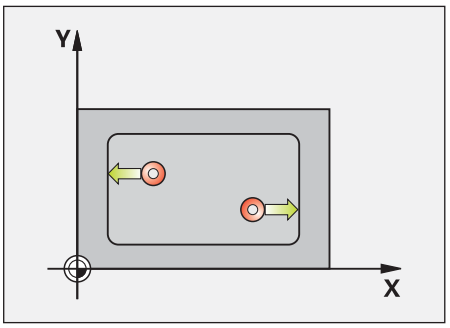


Minimize setup and nonmachining time
– The TNC 320 manages and measures

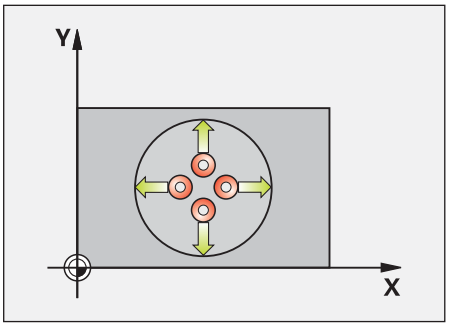
The difference in requirements placed on the classical machine for tool and mold-making and machining centers is becoming ever less distinct. Of course, the TNC 320 is capable of controlling automated manufacturing processes. It has the functions required to manage and measure tools and inspect the machining process. It helps you reduce non-cutting time, increase productivity and improve production quality.

Inspecting workpieces for proper machining and dimensional accuracy
The TNC 320 features a number of measuring cycles for checking the geometry of the machined workpieces. To run the measuring cycles, you simply insert a touch probe from HEIDENHAIN (see page 29) into the spindle in place of a tool. This enables you to

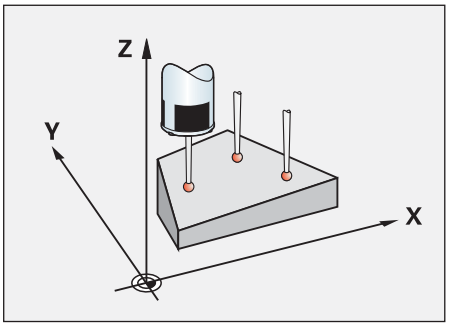
- recognize a workpiece and call the appropriate part program,
- check whether all machining operations were conducted correctly,
- determine infeeds for finishing,
- detect and compensate tool wear,
- ascertain the machining error trend.



Length measurement

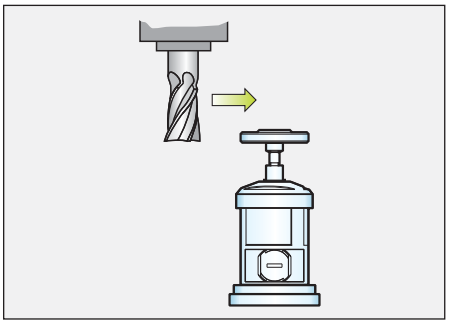


Circular pocket/hole measurement

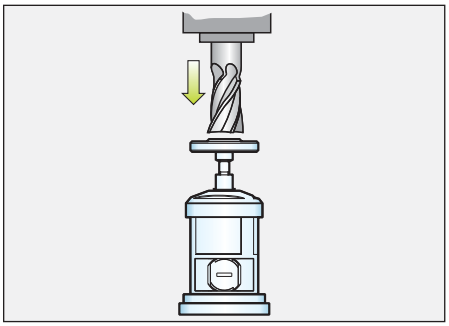


Measuring the angle of a plane

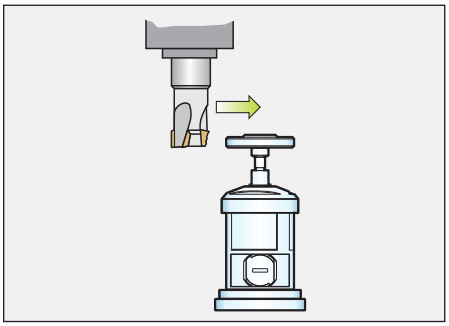
Tool measurement and automatic compensation of tool data
Together with the TT and TL touch probes (see page 30), the TNC 320 can automatically measure tools while they are in the machine. The TNC 320 saves the ascertained values of tool length and radius in the central tool file. By inspecting the tool during machining you can quickly and directly measure wear or breakage to prevent scrap or rework. If the measured deviations lie outside the tolerances, or if the monitored life of the tool is exceeded, the TNC 320 locks the tool and automatically inserts a replacement tool.



Measure tool radius



Measure tool length



Measuring tool wear

Tool management
For machining centers with automatic tool changers, the TNC 320 offers a central tool management function for any number of tools. The tool management is a freely configurable file and can therefore be optimally fitted to your needs. You can even have the TNC 320 manage your tool names. The control prepares the next tool change while the current tool is still cutting. This significantly reduces the non-cutting time required for changing tools.

Economical and productive
– Swivel head and rotary table controlled by the TNC (option)

Many 5-axis operations that at first glance may seem very complex can be reduced to conventional 2-D movements that are simply tilted about one or more rotary axes or wrapped onto a cylindrical surface. The TNC supports you with application-oriented functions to help you write and edit such programs quickly and simply without a CAD/CAM system.

Tilting the working plane*
Programs for contours and holes on inclined surfaces are often very complex and require time-consuming computing and programming work. Here the TNC 320 helps you to save a great deal of programming time.
You program the machining operation as usual in the working plane, for example in X/Y. The machine runs the program in a plane that has been tilted by one or more rotary axes with respect to the main plane.

The PLANE feature makes it easy to define a tilted working plane: You can specify tilted working planes in seven different ways, depending on the information on the workpiece drawing. Clearly arranged support graphics assist you during input.

You can also use the PLANE function to define the positioning behavior for tilting so that there are no unpleasant surprises when the program is run. The settings for defining the positioning behavior are identical for all PLANE functions, making everything that much easier.

Machining cylindrical surfaces*
With the TNC 320 it is quite easy to program contours (which consist of straight lines and arcs) on cylindrical surfaces using rotary and tilting tables: You simply program the contour in a plane as if the cylinder surface were unrolled. You enter a contour in two dimensions—as if in a plane—and the TNC 320 then calculates and machines the corresponding cylindrical contour.

The TNC 320 features three cycles for cylindrical surface machining:
- Slot milling (the slot width is the same as the tool diameter)
- Guide-groove milling (the slot width is greater than the tool diameter)
- Ridge milling

* The machine must be prepared by the machine tool builder for this function.

– Machining any contour slots with trochoidal milling

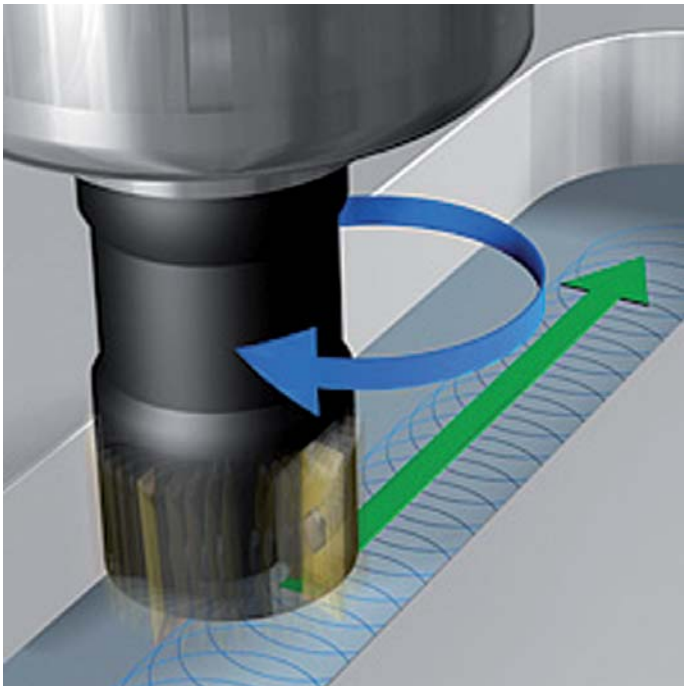
The benefit of trochoidal milling is its ultra-efficient machining of slots of all kinds. The roughing process is a circular motion superimposed on a forward linear motion. This procedure is referred to as trochoidal milling. It is used particularly for milling high-strength or hardened materials, where the high loads placed on the tool and machine usually only permit small infeeds.

With trochoidal milling, on the other hand, large cutting depths are possible since the prevailing cutting conditions do not increase the wear and tear on the tool. On the contrary, the entire length of a plain cutter's cutting edges can be used. This enables you to achieve a greater chip volume per tooth. Circular plunging into the material places less radial force on the tool. This reduces the mechanical load on the machine and prevents vibration.

The slot to be machined is described in a contour subprogram as a contour train. You define the dimensions of the slot and the cutting data in a separate cycle. Any residual material remaining can then easily be removed with a subsequent finishing cut.

The benefits include:

- Engagement of the entire cutter length
- Higher chip volume
- Relief from mechanical load on the machine
- Less vibration
- Integrated finishing of the side wall



Programming, editing, testing
– The TNC 320 opens endless possibilities

The TNC 320 is just as universal in application as it is flexible in machining and programming.

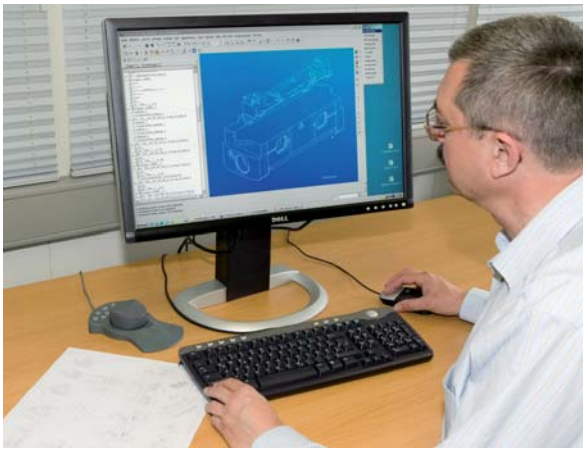
Positioning with Manual Data Input
You can start working with the TNC 320 even before writing a complete part program. Simply machine a part step by step—switching as you want between manual operation and automatic positioning.

Programming at the machine
HEIDENHAIN controls are workshop oriented, which means that they were conceived for programming right at the machine. With **conversational programming** you can forget about memorizing G codes. Instead you use dedicated keys and soft keys to program line segments, circular arcs and cycles. With a keystroke, you initiate a HEIDENHAIN plain-language dialog, and the TNC begins immediately to support you actively in your work. Unambiguous questions and prompts help you enter all the required information.

If you are used to **DIN/ISO programming,** however, the TNC 320 is still the right control for you. It displays soft-key rows dedicated to the most important DIN/ISO commands so that you can enter DIN/ISO programs directly. Or you connect a USB keyboard and use it to write the program.

Whether plain-language prompts, dialog guidance, programming steps or soft keys, all texts are available in numerous languages.

Creating programs offline
The TNC 320 is also well equipped for offline programming. Through its interfaces it can be integrated into networks and connected with programming stations or other data storage devices. The TNC 320 can also run programs that were written in DIN/ISO format.



– Graphic support in any situation

Programming graphics
The two-dimensional programming graphics give you additional security: while you are programming, the TNC 320 draws every entered traverse command on the screen.

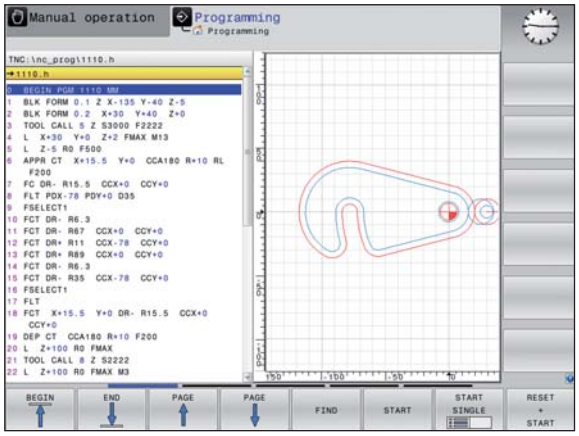
Help graphics
During cycle programming in the plain-language dialog, the TNC shows a separate illustration for each parameter. This makes it easier to understand the function and accelerates programming.

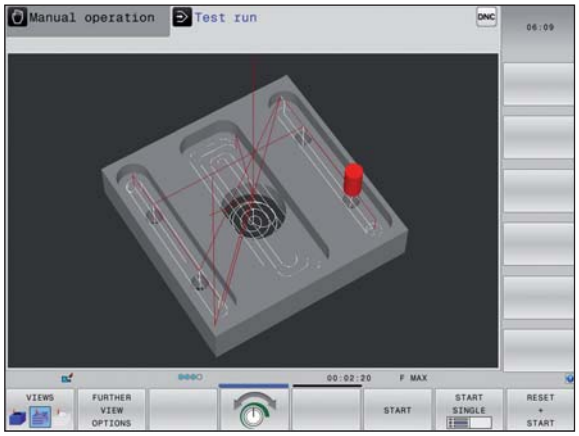
Program verification graphics
To play it safe before running a program, the TNC 320 can graphically simulate the machining of the workpiece. It can display the simulation in the following ways:
- In a plan view with different shades of depth
- In three planes (as in the workpiece drawing)
- In a solid model, 3-D view

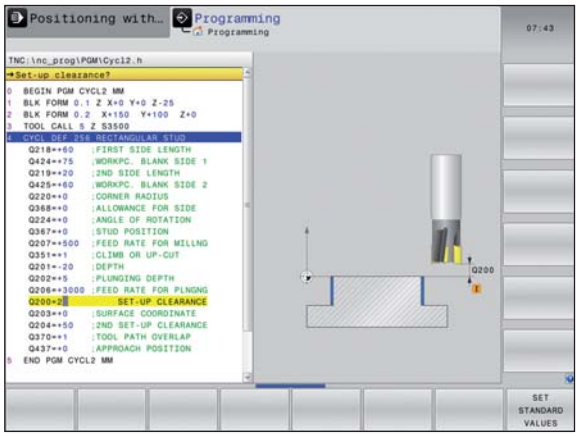
Details can be displayed in magnification. In addition, the TNC 320 indicates the calculated machining time in hours, minutes and seconds.

Program-run graphics
On the TNC 320, you can run the programming graphics or verification graphics even while the workpiece is being machined. Also, it shows a real-time graphic of the machining progress during program run. Coolant spray and protective enclosures usually obstruct any direct view of the actual workpiece. You can get around this with a simple keystroke to see the simulated progress of workpiece machining.

3-D line graphics
The 3-D line graphics display the programmed tool center point path in three dimensions. With the powerful zoom function you can also see the finest details. You should especially use the 3-D line graphics to inspect programs created offline for irregularities before machining, in order to avoid undesirable traces of the machining process on the workpiece, e.g. when points are output incorrectly by the postprocessor.







Programming in the workshop
– Straightforward function keys for complex contours

Programming 2-D contours

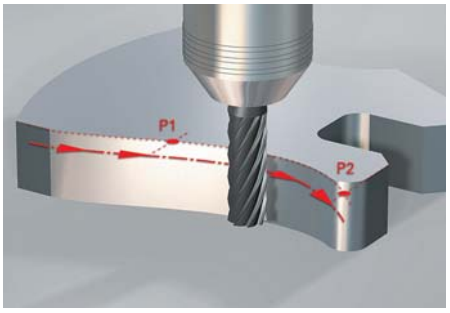
Two-dimensional contours are the bread and butter of the modern machine shop. Here the TNC 320 offers a variety of possibilities.

Programming with path function keys
If contours are dimensioned for NC, which means that the end points are specified in Cartesian or polar coordinates, then you can program them directly with the path function keys.

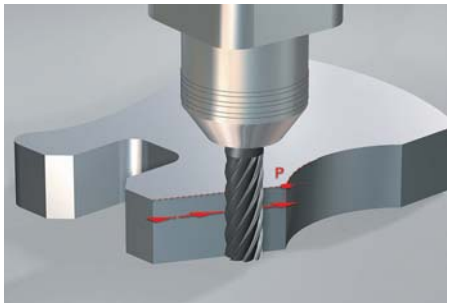
Straight and circular contour elements
To program a line segment, for example, simply press the key for linear traverse. The TNC 320 asks for all information required for a complete programming block, such as target coordinates, feed rate, cutter radius compensation and machine functions. Appropriate path function keys for circular movement, chamfers, and corner rounding simplify your programming. To avoid surface blemishes during approach or departure from the contour, it must be approached smoothly—that is, tangentially.

You simply specify the starting or end point of the contour and the approaching or departing radius of the cutter edge—the control does the rest for you.

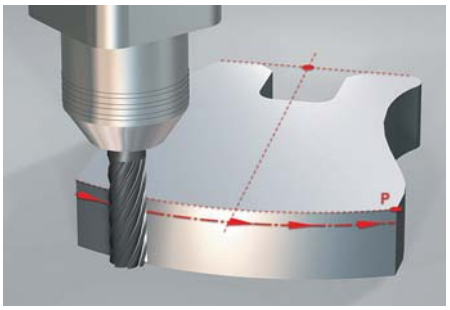
The TNC 320 can look ahead over a radius-compensated contour for up to 99 blocks to watch for back cutting and avoid contour damage such as can occur when roughing a contour with a large tool.



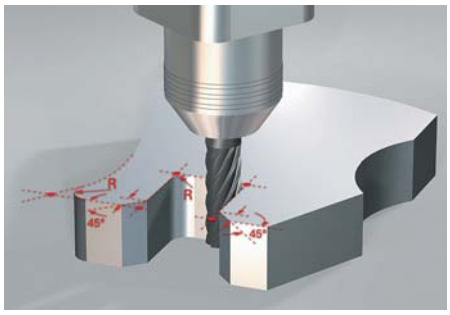
CT
Circular path defined by its end point, with a smooth (tangential) departure from the previous contour element



L
Straight line defined by its end point

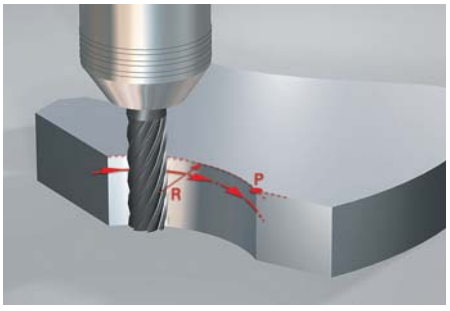


CC C
Circular path defined by its center point, end point, and rotational direction



RND Corner rounding: circular path defined by radius and corner point, with a smooth (tangential) transition to its adjoining contour elements.

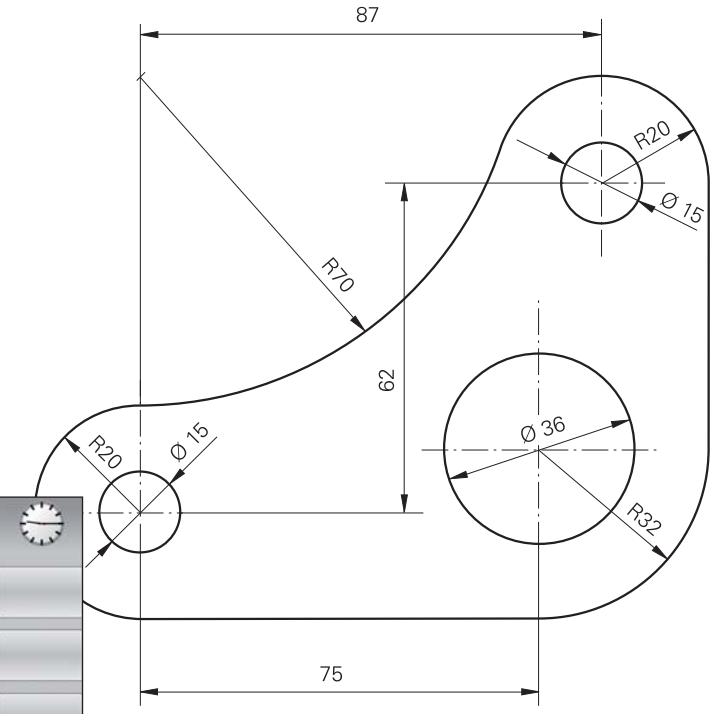
CHF Chamfer: defined by the corner point and the chamfer length

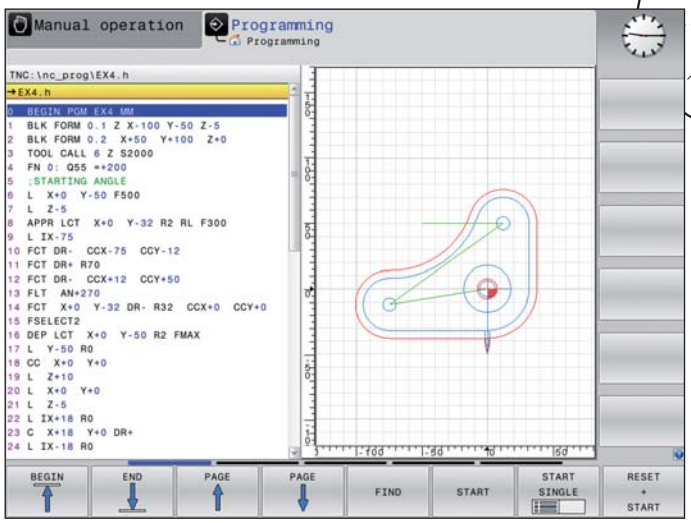


CR
Circular path defined by its radius, end point and rotational direction

– Programming contours unconventionally

FK free contour programming
Not all workpieces are dimensioned for conventional NC programming. Thanks to FK, the control's free contour programming feature, in such cases you simply type in the known data—without first having to convert or calculate your data! It does not matter if individual contour elements are not completely defined as long as the complete contour has been. If the given data result in more than one mathematical solution, the helpful TNC 320 programming graphics offer the possible variants for your selection.





Programming in the workshop
– Field-proven cycles for recurring operations

Comprehensive fixed cycles for milling, drilling and boring

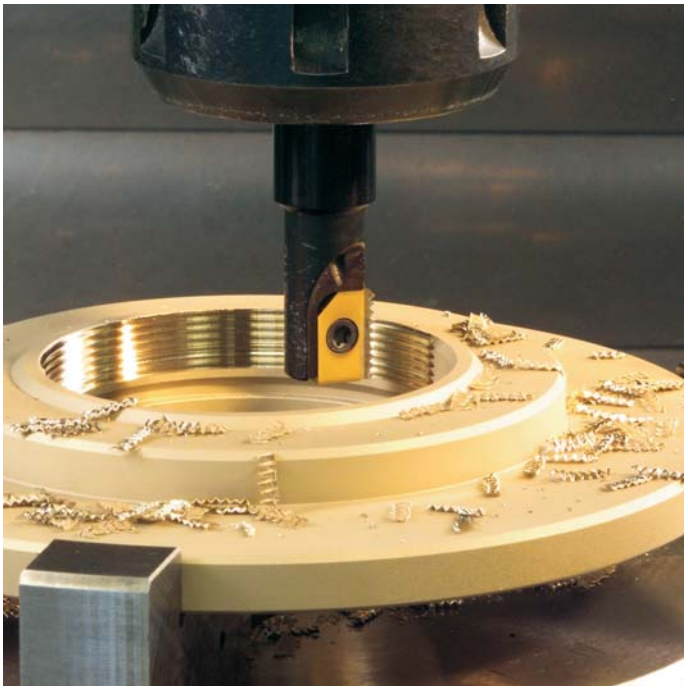
Frequently recurring operations that comprise several working steps are stored in the TNC 320 as cycles. You program them under conversational guidance and are supported by graphics that clearly illustrate the required input parameters.

Standard cycles
Besides the fixed cycles for drilling, tapping (with or without floating tap holder), thread milling, reaming and boring, there are cycles for hole patterns and milling cycles for clearing plane surfaces, and for roughing and finishing pockets, slots and studs.

Cycles for complex contours
Clearing pockets with combined contours is aided greatly by **Subcontour List cycles** (SL). This term is used to identify machining cycles for pilot drilling, roughing and finishing when the contour or subcontours are specified in subroutines. In this way, one contour description can be used for more than one operation using different tools.

Up to twelve **subcontours** can be superimposed for machining. The control automatically calculates the resulting contour and the tool paths for roughing or clearing the surfaces. Subcontours can be pockets or islands. Different components are combined to form a single pocket in which the tool avoids the islands.

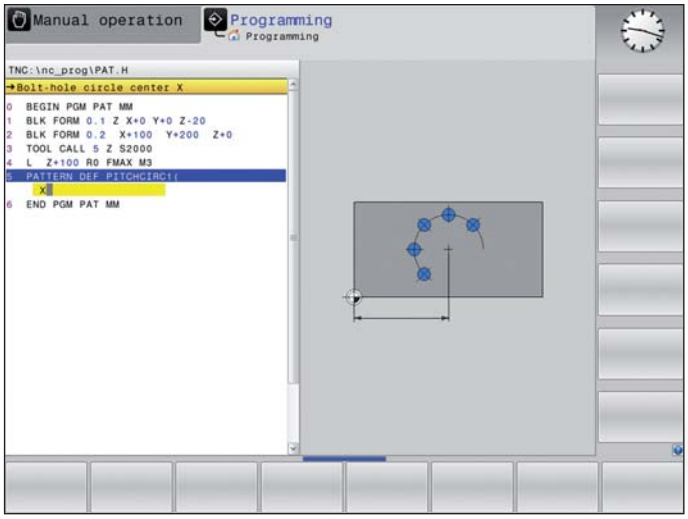
The TNC 320 maintains a **finishing allowance** on the wall and floor surfaces during roughing. When **roughing** with different tools, the control recognizes the material remaining in inside corners so that it can clear it with smaller tools. A separate cycle is used for milling to the finished dimension.



OEM cycles
As original equipment manufacturers (OEMs), machine tool builders can contribute their special manufacturing know-how by designing additional fixed cycles and saving them in the TNC 320. However, the end user can write his own cycles as well. HEIDENHAIN makes this possible with its PC program CycleDesign. CycleDesign enables you to organize the input parameters and soft-key structure of the TNC 320 to suit your own needs.

Simple and flexible programming of machining patterns
Machining positions are often arranged in patterns on the workpiece. With the TNC 320, you can program very diverse machining patterns simply and extremely flexibly—of course with graphic support. You can define as many point patterns as desired with various numbers of points.

3-D machining with parametric programming
With parameter functions you can program simple 3-D geometric figures that can easily be described mathematically. Here you can use the basic arithmetical operations, trigonometric functions, roots, powers, logarithmic functions, parentheses, and logical comparisons with conditional jump instructions. Parametric programming also offers you a simple method of realizing 3-D operations. Of course, parametric programming is also suited for **2-D contours** that cannot be described with line segments or circular arcs, but rather through mathematical functions.



Programming in the workshop
– Reusing programmed contour elements

Coordinate transformation
If you should need a contour that has already been programmed at another position or in a different size, the TNC 320 offers you a simple solution: coordinate transformation.
With coordinate transformation you can, for example, **rotate or mirror** the coordinate system, or **shift the datum**. With a **scaling factor** you can enlarge or reduce contours to respect shrinkage or oversizes.

Program section repeats, subprograms, program calls
Many machining operations repeat themselves either on the same workpiece or on different workpieces. Once you have programmed a detail there is no reason to have to program it again. With its subprogramming feature, the TNC can save you a great deal of programming time.

In **program section repetition**, you label a section of the program and during program run the TNC repeats the section successively as many times as required.

You can mark a program section as a **subprogram** and then call it at any point in the program and as often as you want.

With the **program call** function you can even use a completely separate program at any place in your current program. This gives you convenient access to preprogrammed, frequently needed working steps or contours.

Of course you can also combine these programming techniques.

– Fast availability of all information

Do you have questions on a programming step, but your User's Manual is not at hand? No problem: The TNC 320 numerical control and TNC 320 programming station now feature TNCguide, a convenient help system that can show the user documentation in a separate window.

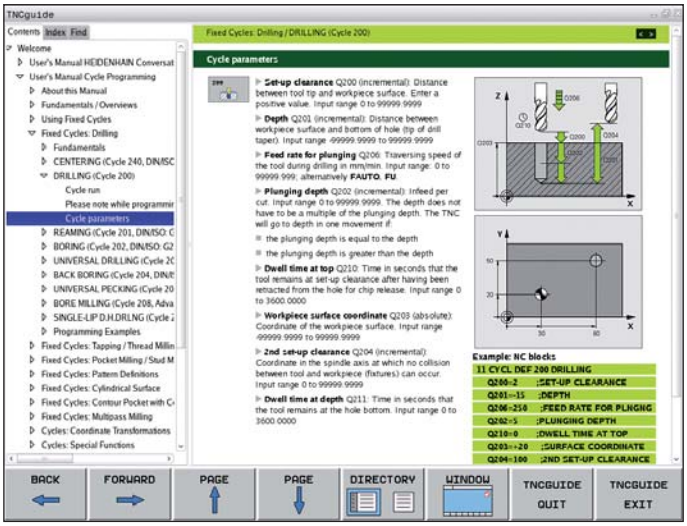
You can activate TNCguide by simply pressing the help key on the TNC keyboard or by clicking any soft key with a cursor in the shape of a question mark. You switch the cursor by simply clicking the help symbol that is always visible on the TNC screen.

TNCguide usually displays the information in the immediate context of the element in question (context-sensitive help). This means that you immediately receive the relevant information. This function is particularly helpful with the soft keys. The method and effect of operation are explained in detail.

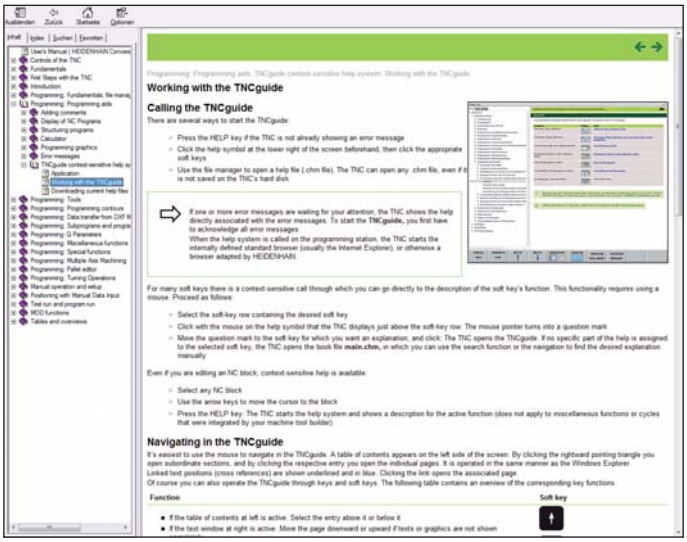
You can download the documentation in the desired language free of charge from the HEIDENHAIN homepage into the corresponding language directory on the TNC hard disk.

The following manuals are available in the help system:

- User's Manual for Conversational Programming
- User's Manual for Cycle Programming
- User's Manual for DIN/ISO Programming
- User's Manual for the TNC 320 Programming Station (only included in the programming station)



TNCguide integrated in the control, e.g. on the TNC 320 ...



… or at the programming station.

Open for communication
– The TNC 320 understands DXF files (option)

Why program complex contours when your drawing is already in DXF format anyway? You can open DXF files directly on the TNC 320 in order to extract contours or machining positions from it. Not only does this save time otherwise spent on programming and testing, but you can also be sure that the finished contour is exactly according to the designer's specifications.

The DXF format—particularly the DXF format supported by the TNC 320—is very widespread, and is supported by all common CAD and graphics programs.

After the DXF file has been loaded onto the TNC from the network or a USB stick, you can open the file in the file manager of the TNC, just like an NC program.

As a rule, DXF files contain multiple layers, with which the designer organizes the drawing. So that as little unnecessary information as possible appears on the screen during selection of the contours, you can hide via mouse click all **excessive layers** contained in the DXF file. An external pointing device is required for this. The TNC can select a contour train even if it has been saved in **different layers.**

The TNC also supports you when **defining the workpiece preset.** The datum of the drawing for a DXF file is not always located where you can use it directly as the workpiece preset, especially when the drawing contains multiple views. For this reason, the TNC has a function with which you can shift the drawing datum to a suitable location simply by clicking an element.

You can define the following locations as reference point:

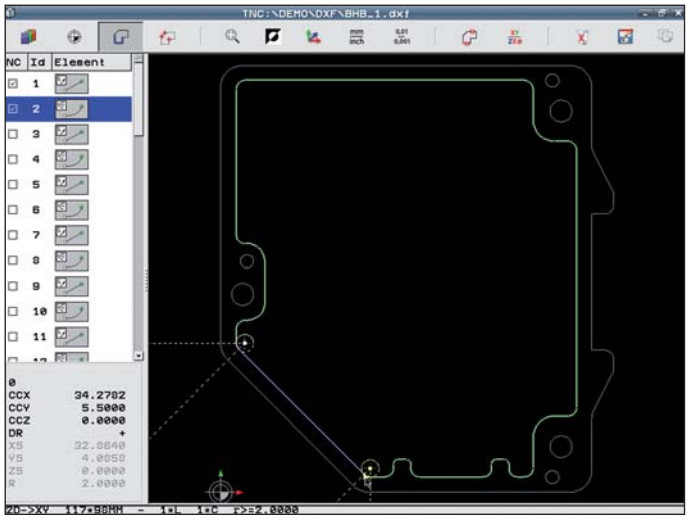
- The beginning, end or mid-point of a line
- The beginning, end or mid-point of a circular arc
- Quadrant transitions or center point of a circle
- Intersection of two lines, regardless of whether it is located inside or outside the programmed segments
- Intersection of a line and a circular arc
- Intersection of a line and a circle

If multiple intersections result between two elements (e.g., between a straight line and a circle), you can select the correct intersection with a mouse click.

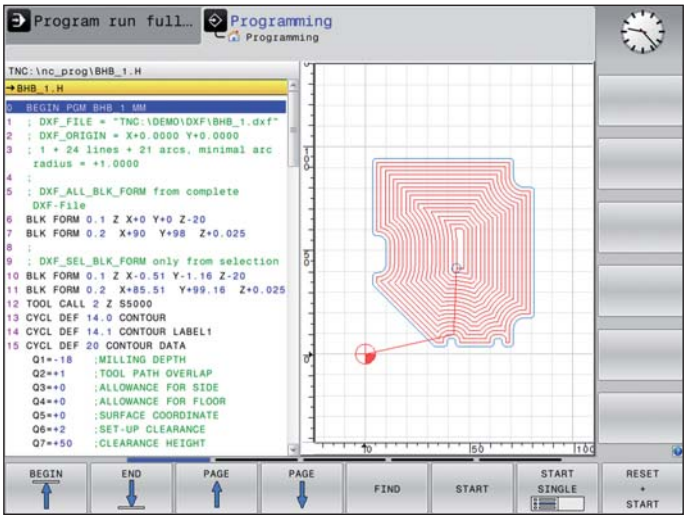
Contour selection is exceptionally user friendly. You select any element by clicking it with the mouse. As soon as you select a second element, the TNC detects your desired direction of machining, and starts the **automatic contour detection.** The TNC automatically selects all clearly identifiable contour elements until the contour closes or branches out. There you click the immediately following contour element. In this way you can define even extensive contours with just a few mouse clicks. If desired you can also shorten, lengthen or interrupt the contour elements.

But you can also select **machining positions** and save them as point files, particularly in order to use drilling positions or starting points for pocket machining. This can be done very easily: Using the mouse, simply select the desired area. In a pop-up window with filter function, the TNC displays all hole diameters that are within the area you have selected. To select the desired hole diameters and restrict the number of hole positions, simply click the corresponding filter symbol to change the filter limits.

A zoom function and various possibilities for settings round out the functionality of the DXF converter. Moreover, you can define the resolution of the contour program to be uploaded in case you want to use it on older TNC controls, or a transition tolerance if occasionally the elements do not quite adjoin.



Contour selection from an imported DXF file



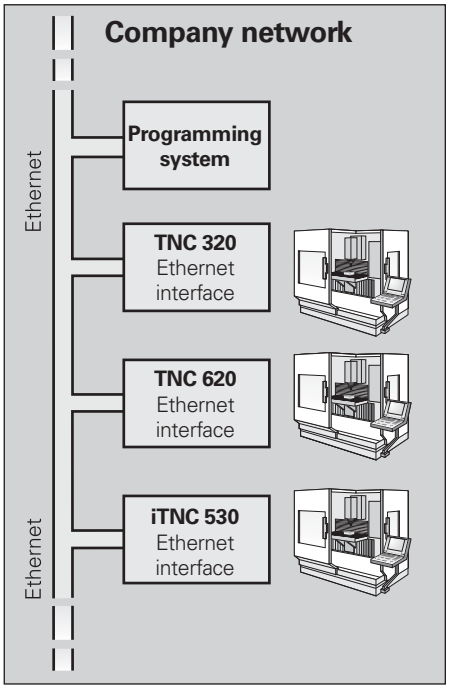
Part program on the basis of the imported DXF file

Open for communication
– Fast data transfer with the TNC

The networked TNC 320
The TNC 320 can be integrated into networks and connected with PCs, programming stations and other data storage devices. Even in its standard version, the TNC 320 features a latest-generation Fast Ethernet interface in addition to its RS-232-C/V.24 data interface. The TNC 320 communicates with NFS servers and Windows networks in TCP/IP protocol without needing additional software. The fast data transfer at rates of up to 1000 Mbit/s guarantees very short transfer times.

The transmitted programs are saved in the internal memory of the TNC 320 and are run from it at high speed.

For well-organized program management on your control, simply place the individual files in directories (folders). You can structure the respective directories through individual subdirectories.



Programs for data transfer
With the aid of the free PC software **TNCremo** from HEIDENHAIN, you can
- transfer remotely stored part programs and tool or pallet tables in both directions,
- make backups.

With the powerful **TNCremoPlus** PC software you can also transfer the screen contents of the control to your PC using the live-screen function.

– Display any file formats on the TNC screen

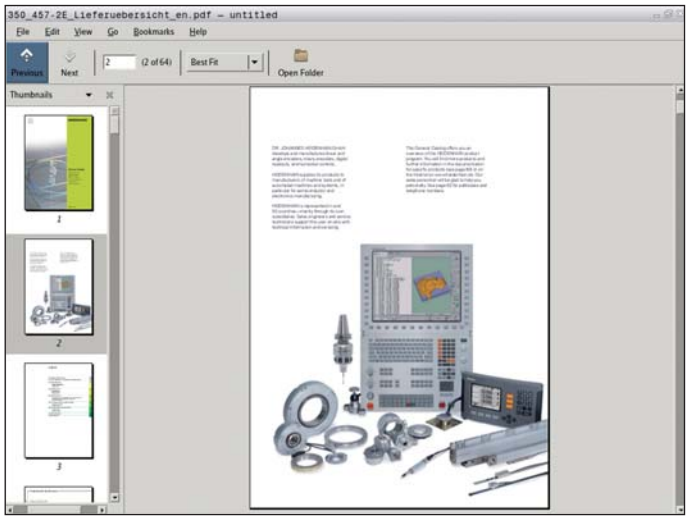
The new integrated PDF viewer enables the user to open PDF files directly on the control. The PDF format is a widely used data format that can be generated out of a great variety of applications. This enables you to easily view work instructions, drawings or other information in the TNC.

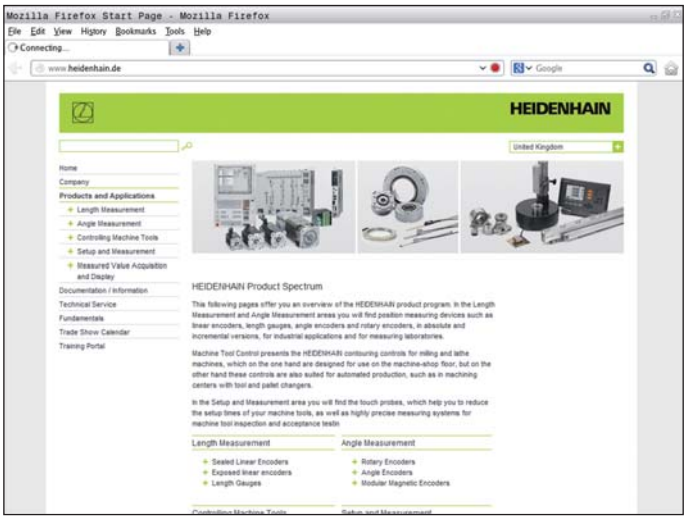
The integrated browser now lets you connect the TNC to the Internet and access it directly from the control.

The following further file formats can also be opened directly on the TNC with corresponding editors, and sometimes edited.

- Text files ending with .txt or .ini
- Graphic files ending with .gif, .bmp, .jpg, .png
- Table files ending with .xls or .csv
- HTML files

An operating panel with an integrated touchpad or an external USB pointing device is required for operation.





Open for communication
– The TNC 320 programming station

Why a programming station?
It's well known that it is easy to create part programs on a TNC 320 at the machine, even while another part is being machined. Nevertheless, it can often happen that short reloading times and other machining tasks hinder any prolonged or concentrated programming work. With the TNC 320 programming station you have the capability to program just as you do at the machine, but away from the noise and distractions of the shop floor.

Creating programs
Programming, testing and optimizing HEIDENHAIN conversational or DIN/ISO programs for the TNC 320 with the programming station substantially reduces machine idle times. And you need not adjust your way of thinking—every keystroke fits, because on the programming station you program on the same keyboard as at the machine.

Testing of programs created offline
Of course you can also test programs that were written on a CAD/CAM system. The various views of the program verification graphics help you to easily spot contour damage and hidden details.

Training with the TNC 320 programming station
Because the TNC 320 programming station is based on the same software as the TNC 320, it is ideally suited for apprentice and advanced training. Programming and program testing function in exactly the same way as they do on the machine. This gives the trainee the experience needed to enable him to safely operate the machine later.

Because the TNC 320 can be programmed in plain language and in DIN/ISO, the TNC 320 programming station can also be used in schools for TNC programming training.

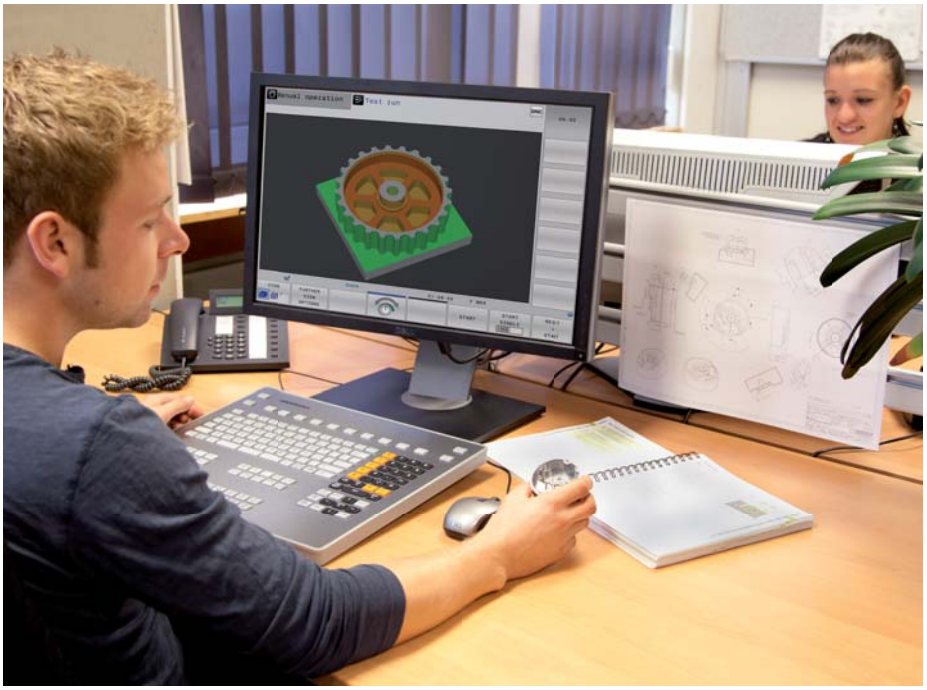
More information about the programming station and a free demo version is available on the Internet at *www.heidenhain.de*. Or simply ask for the *Programming Stations for TNC Controls* CD or brochure.

Your workstation
The programming station software runs on a PC. The PC screen shows you the TNC user interface as on the control, and offers the familiar graphic support. Depending on the version of the programming station, there are several types of possibilities for using it.

The free **demo version** contains all functions of the TNC 320, and permits short programs to be saved. It is programmed over the PC keyboard.

On the version with the TE 520B **TNC operating panel** you then create your programs as always, on a keyboard with the same function keys as on the control of the machine. It also has a PC keyboard for G-code programming, file names and comments.

But you can also work without the TNC operating panel: a **virtual keyboard** simulating the TE appears on the PC screen. It provides the TNC 320's most important dialog initiation keys.



Programming station with TNC operating panel

Workpiece measurement
– Setup, presetting and measuring with touch trigger probes

Workpiece touch probes* from HEIDENHAIN help you to reduce costs in the workshop and in series production: Together with the TNC 320, touch probes can automatically perform setup, measuring and inspection functions.

The stylus of a TS touch trigger probe is deflected upon contact with a workpiece surface. At that moment the TS generates a trigger signal that, depending on the model, is transmitted either by cable or via infrared or radio transmission to the control. An SE transmitter/receiver unit is required for the radio and infrared signal transmission.

The touch probe* is inserted directly into the machine tool spindle. It can be equipped with various shanks depending on the machine. The ruby ball tips are available in several diameters, and the styli in different lengths.

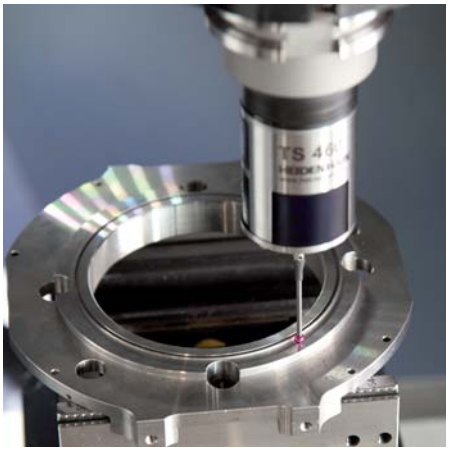
* The touch probes must be interfaced to the TNC 320 by the machine tool builder.

Touch probes with **cable connection for signal transmission** for machines with manual tool change and for grinding machines and lathes:

TS 260 – New generation, axial or radial cable

Touch probe with **radio or infrared signal transmission** for machines with automatic tool change:

TS 460 – New generation standard touch probe for radio and infrared transmission, with compact dimensions
TS 444 – Battery-free voltage supply through integrated air turbine generator over central compressed air, for infrared transmission, with compact dimensions
TS 740 – High probing accuracy and repeatability, low probing force, with infrared transmission



TS 260

TS 460

SE 660

More information about workpiece touch probes is available on the Internet at *www.heidenhain.de* or in the *Touch Probes* brochure.

Tool measurement
– Measuring length, radius and wear directly in the machine

The tool is of course a decisive factor in ensuring a consistently high level of production quality. This means that an exact measurement of the tool dimensions and periodic inspection of the tool for wear and breakage, as well as the shape of each tooth, are necessary. HEIDENHAIN offers the TT trigger tool touch probes as well as the non-contacting TL Nano and TL Micro laser systems for tool measurement.

The systems are installed directly in the machine's workspace, where they permit tool measurement before machining as well as during interruptions.

The **TT tool touch probes** measure the tool length and radius. When probing the tool, either while rotating or at standstill (such as for measuring individual teeth), the contact plate is deflected and a trigger signal is transmitted to the TNC 320.

The **TT 160** uses signal transmission by cable, whereas the **TT 460** operates with wireless signal transmission over radio or an infrared beam. It is therefore particularly suitable for use on rotary and tilting tables.

The **TL Nano** and **TL Micro laser systems** are available for various maximum tool diameters. Using a laser beam, they probe the tool without contact, and can detect form deviations of individual teeth along with the tool length and radius.

TL Micro

TT 460

More information about tool touch probes is available on the Internet at *www.heidenhain.de* or in the *Touch Probes* brochure.

Positioning with the electronic handwheel
– Delicate axis traverse

To set up the workpiece you can use the direction keys to move the machine axes manually. A simpler and more sensitive way, however, is to use the electronic handwheels from HEIDENHAIN.

You can move the axis slide through the feed motors in direct relation to the rotation of the handwheel. For delicate operations you can set the transmission ratio to certain preset distances per handwheel revolution.

Panel-mounted handwheels
The HR 130 and HR 150 panel-mounted handwheels from HEIDENHAIN can be integrated in the machine operating panel or mounted at another location on the machine. An adapter permits connection of up to three HR 150 electronic handwheels.

Portable handwheels
The portable HR 410, HR 520 and HR 550 handwheels are particularly helpful for when you have to work close to the machine's working space. The axis keys and certain functional keys are integrated in the housing. In this way you can switch axes and set up the machine at any time—regardless of where you happen to be standing. The HR 520 and HR 550 handwheels feature an integrated display for user-friendly remote operation of the control. As a wireless handwheel, the HR 550 is ideal for use on large machine tools. If you no longer need the handwheel, just attach it to the machine somewhere by its built-in magnets.

The HR 520 and HR 550 provide the following functions:

- Traverse distance per revolution can be set
- Display for operating mode, actual position value, programmed feed rate and spindle speed, error messages
- Override potentiometer for feed rate and spindle speed
- Selection of axes via keys or soft keys
- Keys for continuous traverse of the axes
- Emergency stop button
- Actual position capture
- NC start/stop
- Spindle on/off
- Soft keys for machine functions defined by the machine tool builder

HR 550

Overview
– User functions

User functions	Default	Option	
Short description	•		Basic version: 3 axes plus closed-loop spindle
		○	1st additional axis for 4 axes plus open-loop or closed-loop spindle
		○	2nd additional axis for 5 axes plus open-loop or closed-loop spindle
Program entry	•		HEIDENHAIN conversational
	•		DIN/ISO via soft keys or via USB keyboard
Position entry	•		Nominal positions for lines and arcs in Cartesian coordinates or polar coordinates
	•		Incremental or absolute dimensions
	•		Display and entry in mm or inches
Tool compensation	•		Tool radius in the working plane and tool length
	•		Radius-compensated contour look-ahead for up to 99 blocks (M120)
Tool tables	•		Multiple tool tables with any number of tools
Cutting data	•		Automatic calculation of spindle speed, cutting speed, feed per tooth and feed per revolution
Constant contour speed	•		Relative to the path of the tool center
	•		Relative to the tool's cutting edge
Parallel operation	•		Creating a program with graphical support while another program is being run
Rotary table machining		8	Programming of cylindrical contours as if in two axes
		8	Feed rate in distance per minute
Contour elements	•		Straight line
	•		Chamfer
	•		Circular path
	•		Circle center
	•		Circle radius
	•		Tangentially connecting circular arc
	•		Corner rounding
Approaching and departing the contour	•		Via straight line: tangential or perpendicular
	•		Via circular arc
FK free contour programming	•		FK free contour programming in HEIDENHAIN conversational format with graphic support for workpiece drawings not dimensioned for NC
Program jumps	•		Subprograms
	•		Program section repeats
	•		Calling any program as a subprogram
Fixed cycles	•		Cycles for drilling, pecking, reaming, boring, counterboring, conventional and rigid tapping
	•		Cycles for milling internal and external threads
	•		Rectangular and circular pockets
	•		Face milling
	•		Cycles for clearing level and inclined surfaces
	•		Multioperation machining of straight and circular slots
	•		Multioperation machining of rectangular and circular pockets
	•		Cartesian and polar point patterns
	•		Contour train, contour-parallel contour pocket
	•		OEM cycles (special cycles developed by the machine tool builder) can be integrated
	•		Engraving cycle: Engrave text or numbers in a straight line or on an arc

User functions	Default	Option	
Coordinate transformation	•	8	Datum shift, rotation, mirror image, scaling factor (axis-specific) Tilting the working plane, PLANE function
Q parameters Programming with variables	• • • • •		Mathematical functions =, +, –, *, /, sin α, cos α, tan α, arc sin, arc cos, arc tan, a^n, e^n, ln, log, $\sqrt{a}$, $\sqrt{a^2 + b^2}$ Logical operations (=, = /, <, >) Calculating with parentheses Absolute value of a number, constant π, negation, truncation of digits before or after the decimal point Functions for calculation of circles
Programming aids	• • • • • •		Calculator Complete list of all current error messages Context-sensitive help function for error messages TNCguide: The integrated help system. User information available directly on the TNC 320 Graphic support for programming cycles Comment and structure blocks in the NC program
Teach-In	•		Actual positions can be transferred directly into the NC program
Program verification graphics Display modes	• • •		Graphic simulation before a program run, even while another program is running Plan view / projection in 3 planes / 3-D view, also in tilted working plane/3-D line graphics Magnification of details
Programming graphics	•		In the Programming and Editing mode, the contour of the NC blocks is drawn on screen while the blocks are being entered (2-D pencil-trace graphics), even while another program is running
Program-run graphics Display modes	• •		Graphic simulation during real-time machining Plan view / projection in 3 planes / 3-D view
Machining time	• •		Calculation of machining time in the Test Run operating mode Display of the current machining time in the Program Run operating modes
Returning to the contour	• •		Mid-program startup in any block in the program, returning the tool to the calculated nominal position to continue machining Program interruption, contour departure and return
Datum management	•		For saving any reference points
Datum tables	•		Several datum tables for storing workpiece-related datums
Touch probe cycles	• • • •		Calibrating the touch probe Workpiece misalignment, compensating Datum setting, manual or automatic Automatic tool and workpiece measurement
Parallel secondary axes	• • •		Compensating movement in the secondary axes U, V, W through the principal axes X, Y, Z Including movements of parallel axes in the position display of the associated principal axis (sum display) Defining the principal and secondary axes in the NC program makes it possible to run programs on different machine configurations
Conversational languages	•		English, German, Czech, French, Italian, Spanish, Portuguese, Swedish, Danish, Finnish, Dutch, Polish, Hungarian, Russian (Cyrillic), Chinese (Traditional/Simplified), Slovenian, Slovak, Norwegian, Korean, Turkish, Romanian

Overview
– Options
– Accessories

Option	Option number	As of NC software 771851-	ID	Remark
Additional axis	–	01	–	• 1st additional axis for 4 axes plus open-loop or closed-loop spindle • 2nd additional axis for 5 axes plus open-loop spindle
Software option 1	**8**	01	536164-01	**Rotary table machining** • Programming of cylindrical contours as if in two axes • Feed rate in distance per minute
		01		**Interpolation** • Circular in 3 axes with tilted working plane **Coordinate transformation** • Tilting the working plane • PLANE function
HEIDENHAIN DNC	**18**	01	526451-01	Communication with external PC applications over COM component
DXF Converter	**42**	01	526450-01	Load and convert DXF contours

Accessories	
Electronic handwheels	• One **HR 410** portable handwheel • One **HR 520** portable handwheel with display or • One **HR 550** portable wireless handwheel with display or • One **HR 130** panel-mounted handwheel or • Up to three **HR 150** panel-mounted handwheels through an HRA 110 handwheel adapter
Workpiece measurement	• **TS 260:** Touch trigger probe with cable connection or • **TS 460:** Touch trigger probe with radio or infrared transmission or • **TS 444:** Touch trigger probe with infrared transmission or • **TS 640:** Touch trigger probe with infrared transmission or • **TS 740:** Touch trigger probe with infrared transmission
Tool measurement	• **TT 160:** 3-D touch trigger probe or • **TT 460:** 3-D touch trigger probe with radio or infrared transmission • **TL Nano:** Laser system for contact-free workpiece measurement or • **TL Micro:** Laser system for contact-free workpiece measurement
Software for PCs	• **TeleService:** Software for remote diagnostics, monitoring, and operation • **CycleDesign:** Software for creating your own cycle structure • **TNCremo:** Software for data transfer—free of charge • **TNCremoPlus:** Software for data transfer with live-screen function
Programming station	Control software for PCs for programming, archiving, and training • Single-station license with original control keyboard • Single-station license with virtual keyboard • Network license with virtual keyboard • Demo version (operated via virtual keyboard or PC keyboard—free of charge)

– Specifications

Specifications	
Components	• Main computer with TNC keyboard and integrated 15.1-inch TFT color flat-panel display with soft keys
Operating system	• HEROS 5 real-time operating system for machine control
Memory	• 1.8 GB (on compact flash memory card CFR)
Input resolution and display step	• Linear axes: up to 0.1 µm • Angular axes: to 0.0001°
Input range	• Maximum 99999.999 mm or 99999.999°
Interpolation	• Linear in 4 axes • Circular in 2 axes • Circular in 3 axes with tilted working plane • Helical: superimposition of circular and straight paths
Block processing time	• 3 ms (3-D straight line without radius compensation)
Axis feedback control	• Position resolution: Signal period of the position encoder/1024 • Cycle time of position controller: 3 ms
Range of traverse	• Maximum 100 m
Spindle speed	• Maximum 100 000 rpm (analog speed command signal)
Error compensation	• Linear and nonlinear axis error, backlash, reversal spikes during circular movements, thermal expansion • Static friction
Data interfaces	• RS-232-C/V.24 max. 115 Kbps • Extended data interface with LSV2 protocol for remote operation of the TNC 320 over the data interface with the HEIDENHAIN software TNCremo or TNCremoPlus • Gigabit Ethernet interface 1000BASE-T • 3 x USB (1 x front USB 2.0; 2 x rear USB 3.0)
Diagnostics	• Fast and simple troubleshooting through integrated diagnostic aids
Ambient temperature	• Operation: 0 °C to +45 °C • Storage: –20 °C to +60 °C

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 8669 31-0
FAX +49 8669 5061
E-mail: info@heidenhain.de

www.heidenhain.de

Vollständige und weitere Adressen siehe www.heidenhain.de
For complete and further addresses see www.heidenhain.de

DE **HEIDENHAIN Vertrieb Deutschland**
83301 Traunreut, Deutschland
☏ 08669 31-3132
FAX 08669 32-3132
E-Mail: hd@heidenhain.de

HEIDENHAIN Technisches Büro Nord
12681 Berlin, Deutschland
☏ 030 54705-240

HEIDENHAIN Technisches Büro Mitte
07751 Jena, Deutschland
☏ 03641 4728-250

HEIDENHAIN Technisches Büro West
44379 Dortmund, Deutschland
☏ 0231 618083-0

HEIDENHAIN Technisches Büro Südwest
70771 Leinfelden-Echterdingen, Deutschland
☏ 0711 993395-0

HEIDENHAIN Technisches Büro Südost
83301 Traunreut, Deutschland
☏ 08669 31-1345

AR **NAKASE SRL.**
B1653AOX Villa Ballester, Argentina
www.heidenhain.com.ar

AT **HEIDENHAIN Techn. Büro Österreich**
83301 Traunreut, Germany
www.heidenhain.de

AU **FCR Motion Technology Pty. Ltd**
Laverton North 3026, Australia
E-mail: vicsales@fcrmotion.com

BE **HEIDENHAIN NV/SA**
1760 Roosdaal, Belgium
www.heidenhain.be

BG **ESD Bulgaria Ltd.**
Sofia 1172, Bulgaria
www.esd.bg

BR **DIADUR Indústria e Comércio Ltda.**
04763-070 – São Paulo – SP, Brazil
www.heidenhain.com.br

BY **GERTNER Service GmbH**
220026 Minsk, Belarus
www.heidenhain.by

CA **HEIDENHAIN CORPORATION**
Mississauga, OntarioL5T2N2, Canada
www.heidenhain.com

CH **HEIDENHAIN (SCHWEIZ) AG**
8603 Schwerzenbach, Switzerland
www.heidenhain.ch

CN **DR. JOHANNES HEIDENHAIN (CHINA) Co., Ltd.**
Beijing 101312, China
www.heidenhain.com.cn

CZ **HEIDENHAIN s.r.o.**
102 00 Praha 10, Czech Republic
www.heidenhain.cz

DK **TP TEKNIK A/S**
2670 Greve, Denmark
www.tp-gruppen.dk

ES **FARRESA ELECTRONICA S.A.**
08028 Barcelona, Spain
www.farresa.es

FI **HEIDENHAIN Scandinavia AB**
02770 Espoo, Finland
www.heidenhain.fi

FR **HEIDENHAIN FRANCE sarl**
92310 Sèvres, France
www.heidenhain.fr

GB **HEIDENHAIN (G.B.) Limited**
Burgess Hill RH15 9RD, United Kingdom
www.heidenhain.co.uk

GR **MB Milionis Vassilis**
17341 Athens, Greece
www.heidenhain.gr

HK **HEIDENHAIN LTD**
Kowloon, Hong Kong
E-mail: sales@heidenhain.com.hk

HR Croatia → **SL**

HU **HEIDENHAIN Kereskedelmi Képviselet**
1239 Budapest, Hungary
www.heidenhain.hu

ID **PT Servitama Era Toolsindo**
Jakarta 13930, Indonesia
E-mail: ptset@group.gts.co.id

IL **NEUMO VARGUS MARKETING LTD.**
Tel Aviv 61570, Israel
E-mail: neumo@neumo-vargus.co.il

IN **HEIDENHAIN Optics & Electronics India Private Limited**
Chetpet, Chennai 600 031, India
www.heidenhain.in

IT **HEIDENHAIN ITALIANA S.r.l.**
20128 Milano, Italy
www.heidenhain.it

JP **HEIDENHAIN K.K.**
Tokyo 102-0083, Japan
www.heidenhain.co.jp

KR **HEIDENHAIN Korea LTD.**
Gasan-Dong, Seoul, Korea 153-782
www.heidenhain.co.kr

MX **HEIDENHAIN CORPORATION MEXICO**
20235 Aguascalientes, Ags., Mexico
E-mail: info@heidenhain.com

MY **ISOSERVE SDN. BHD.**
43200 Balakong, Selangor
E-mail: isoserve@po.jaring.my

NL **HEIDENHAIN NEDERLAND B.V.**
6716 BM Ede, Netherlands
www.heidenhain.nl

NO **HEIDENHAIN Scandinavia AB**
7300 Orkanger, Norway
www.heidenhain.no

PH **Machinebanks` Corporation**
Quezon City, Philippines 1113
E-mail: info@machinebanks.com

PL **APS**
02-384 Warszawa, Poland
www.heidenhain.pl

PT **FARRESA ELECTRÓNICA, LDA.**
4470 - 177 Maia, Portugal
www.farresa.pt

RO **HEIDENHAIN Reprezentanţă Romania**
Braşov, 500407, Romania
www.heidenhain.ro

RS Serbia → **BG**

RU **OOO HEIDENHAIN**
115172 Moscow, Russia
www.heidenhain.ru

SE **HEIDENHAIN Scandinavia AB**
12739 Skärholmen, Sweden
www.heidenhain.se

SG **HEIDENHAIN PACIFIC PTE LTD.**
Singapore 408593
www.heidenhain.com.sg

SK **KOPRETINA TN s.r.o.**
91101 Trencin, Slovakia
www.kopretina.sk

SL **NAVO d.o.o.**
2000 Maribor, Slovenia
www.heidenhain.si

TH **HEIDENHAIN (THAILAND) LTD**
Bangkok 10250, Thailand
www.heidenhain.co.th

TR **T&M Mühendislik San. ve Tic. LTD. ŞTİ.**
34728 Ümraniye-Istanbul, Turkey
www.heidenhain.com.tr

TW **HEIDENHAIN Co., Ltd.**
Taichung 40768, Taiwan R.O.C.
www.heidenhain.com.tw

UA **Gertner Service GmbH Büro Kiev**
01133 Kiev, Ukraine
www.heidenhain.ua

US **HEIDENHAIN CORPORATION**
Schaumburg, IL 60173-5337, USA
www.heidenhain.com

VE **Maquinaria Diekmann S.A.**
Caracas, 1040-A, Venezuela
E-mail: purchase@diekmann.com.ve

VN **AMS Co. Ltd**
HCM City, Vietnam
E-mail: davidgoh@amsvn.com

ZA **MAFEMA SALES SERVICES C.C.**
Midrand 1685, South Africa
www.heidenhain.co.za

Zum Abheften hier falzen! / *Fold here for filing!*

1113511-20 · 15 · 9/2014 · F&W · Printed in Germany